International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN (P): 2249-6890; ISSN (E): 2249-8001
Vol. 7, Issue 6, Dec 2017, 213-222

NUMERICAL ANALYSIS OF FLOW OVER NACA0012 AT FIXED MACH NUMBER, USING COMPUTATIONAL FLUID DYNAMICS

PALLAVI PATURU[1], GANDHI MALLELA[2], S. JEBAROSE JULIYANA[3], A. DIVYA SADHANA[4] & M. KOMALESWARAO[5]

[1,2,3,4]Assistant Professor, Department of Mechanical Engineering, Vel Tech Dr. RR & Dr. SR University, Chennai, Tamil Nadu, India

[5]Assistant Professor, Department of Mechanical Engineering, JNTUK-University College of Engineering, Vizianagaram, Andhra Pradesh, India

ABSTRACT

In this present work, we carry out a numerical analysis of lift and drag performances of NACA0012 airfoil at various angles of attack, by using computational fluid dynamic software. We can also utilize wind tunnel testing setup, to determine lift and drag force. In this investigation process, the design model has to be placed in the test section. This process is more laborious and expensive, than computational fluid dynamic techniques. This analytical method can be experimental, by investigation test. This numerical analysis of the two dimensional subsonic flow over a NACA0012 a airfoil at various angle of attack ranging from 2 to 10 degree (at a difference of 2 degree) at fixed Mach number 0.44 is presented. For this analysis we use k-ω SST turbulence model. The final result of analysis of CFD simulation shows the similarity to the exiting finding in the modern literature. Through this research we are proposing a reliable alternative experimental method for determining co-efficiency of drag and lift.

KEYWORDS: k-ω SST Turbulence, Lift, Drag & CFD.

Received: Oct 05, 2017**; Accepted:** Oct 27, 2017**; Published:** Nov 08, 2017**; Paper Id.:** IJMPERDDEC201723

Original Article

INTRODUCTION

It's a fact of general experience that bodies in motion through a fluid convergence with a resultant force which, in most cases is mainly motion of the resistance. A class of body exists, in any case, for which the segment of the resultant force typical to the bearing of the movement is ordinarily more prominent than the part opposing the movement, also, the likelihood of the flight of a plane relies upon the utilization of the body of this class for wing structure.

An Airfoil is such an aerodynamic shape, to the point that when it moves through the air, the air is split and goes above and underneath the wing. The wing's upper surface is formed so the air hurrying over the best accelerates and extends. This abatement is the gaseous tension over the wing. The air streaming at a later place the wing moves in a generally straighter line, so its speed and pneumatic stress continue as before. Since high pneumatic stress dependably advances toward low gaseous tension, the air at a later place the wing pushes upward toward the air over the wing. The wing is in the inside, and the entire wing is lifted.

Airfoil profile is the imperative parameter for wing plan since wing productivity increments relying upon airfoil profile, so there are a ton of concentrates over the airfoil profile as numerical in the writing. In any case,

those take much time and monetary and at whatever point we need to change a parameter about our examination, it is extremely troublesome on account of time and financial. Good observer can consider quick and effectively, on account of computational liquid progression (CFD) programs. These projects can give right outcomes as trial strategies. Additionally, CFD projects can be contributed, as respects time and quicker, as per test strategies. NACA airfoil sorts were researched in the writing. For the most part, there is a considerable measure of the agent's contemplated lift and drag exhibitions, of NACA airfoil.

Angle of Attack (AOA)

Approach is the point between the approaching air (α) or relative wind and a reference line (V), on the plane or wing. As the nose of the wing turns up, Angle of attack increases and lift force also increases. Drag goes up additionally, however not as fast as lift. Amid remove a plane develops to a specific speed and afterward the pilot turns the plane that is, the pilot controls the controls with the manipulates that the nose of the plane comes up and, at some Angle of attack the wings create enough lift to take the plane into the air.

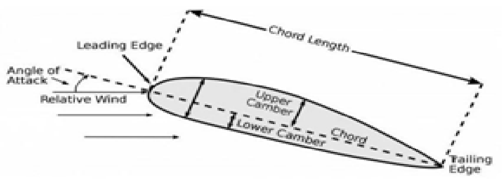

Figure 1: Nomenclature of an Airfoil of Regular Surfaces

METHOD'S

Computational fluid dynamics is a numerical strategy used to recreate physical problems with utilization of governing equations. This technique can be utilized to explore design approaches without making a physical model and can be a profitable instrument to comprehend applied properties of new mechanical design. By utilizing a reenactment as opposed to doing lab tests, one may gain comes about quicker and with less cost. An essential perspective in the utilization of CFD is to comprehend the rearrangements in programming, and know the restrictions in the registered outcomes. In spite of the fact that the CFD programming utilizes surely understood overseeing conditions, serious improvements are made regarding grid and speaking to geometries.

In this model, NACA0012 the all around recorded aerofoil, from the four digit arrangement of NACA aerofoils is used. The NACA0012 aerofoil is symmetrical; the 00 demonstrated that it has no camber. The 12 demonstrates that, the aerofoil has a 12% thickness to harmony length proportion. Reynolds number for the reproductions was Re = 3×10^5, with a specific end goal, to approve the present re-enactment.

Approach

Following is the steps employed, to carry through the CFD simulation:

1) Geometric model 2) Meshing 3) Boundary conditions 4) FLUENT solving

Preparing Geometric Model

NACA 0012 symmetric aerofoil geometry was gained as co-ordinate vertices i.e. writings text files and imported

into the ANSYS FLUENT. A couple of changes were made to this to remedy the geometry and make it substantial as a CFD demonstrate. Familiar is basic during the time spent doing the CFD analysis; it makes the workplace where the protest is mimicked. An essential part in this is making the work encompassing the question. This should be stretched out every which way to get the physical properties of the encompassing liquid for this situation moving air. The work and edges should likewise be gathered keeping in mind the end goal to define the fundamental limit (boundary) conditions successfully.

Right off the bat we need to import the coordinates of aerofoil and make the curve, the 2D analysis sort is utilized and dispatch the outline model made. At that point we have to make the surface to the curve then the aerofoil is produced. We have to make the cross section surface we will utilize once we start to determine limit (boundary) conditions. We will start by making an organize framework at the tail of the aerofoil this will enable us to make the geometry for the C-mesh space by utilizing sketcher tool stash and measurement tool kit. Next, we have to make a surface from this portray. The last advance of making the C-mesh is making a surface between the limit and the aerofoil by utilizing Boolean operations. In the last advance of making the geometry, we will part up the new surface into 4 quadrants; this will be valuable for cross section the geometry. The geometry is done. Spare the task and close the plan modeler, as we are currently we are prepared to make the work for the reproduction.

Generate Meshing

A domain comprising of 2 squares and 1 crescent encompasses the NACA 0012 symmetrical aerofoil. The mesh is developed to be fine at locales near the aerofoil and with high vitality, and coarser more distant far from the aerofoil. For this aerofoil an organized quadratic mesh was utilized. Because of constraints in the FLUENT programming, the work must be fine additionally in specific districts a long way from the aerofoil. A fine work infers a higher number of counts which thus influences the reproduction to utilize longer time to wrap up. For the NACA aerofoil, the matrix disseminated with an expanding separation between hubs, beginning from little sizes from the main edge. From the purpose of max thickness on the aerofoil to the trailing edge a significantly number of focuses is disseminated on the aerofoil surface.

Grid Convergence

Two distinct cross sections are made to the CFD reproductions. This is to test the lattice joining, how the adjustment in number of cells, and henceforth additionally cell estimate, influence the final product.

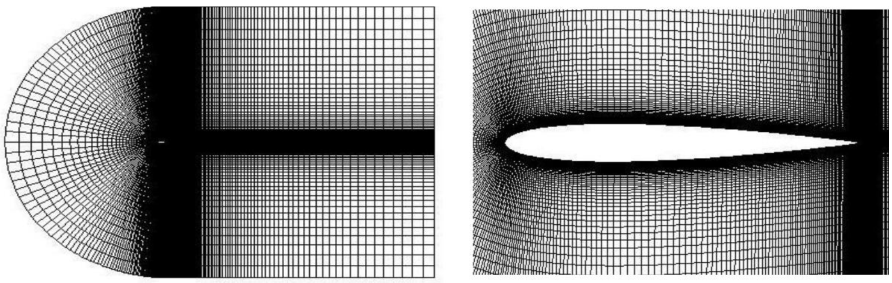

Figure 2: Mesh Generated Around NACA0012 Airfoil (left) and Detail Closed to the Airfoil (Right)

Setting Boundary Conditions

Giving properties to the diverse geometries is imperative to influence the reproduction to work. For this situation, the work limits were offered set to the x and y speed parts, and the end limit the property "pressure far-field" to mimic the

zero gage weight. The aerofoil itself is given as divider properties. In this issue considers stream around an airfoil at various approach. For this, we take some underlying data sources and limit condition for our concern which is appeared in the table beneath.

Table 1: Operating Parameters of Inputs Values

Sl. No.	Input	Values
1	Velocity of flow	0.44 Mach
2	Operating temperature	300 K
3	Operating pressure	0 Pa
4	Model	k-ω SST Turbulence Model
5	Density of fluid	1.225 kg/m^3
6	Viscosity	1.79E-005 kg/m-s
7	Length	1 unit
8	AOA (in degrees)	2,4,6,8 and 10
9	Fluid	Air as ideal gas

FLUENT Solving

The geometry and mesh were imported into FLUENT, and the framework and condition properties set "PRESTO" and “Twofold exactness" is chosen as framework parameters, guaranteeing satisfactory precision. Familiar has single exactness as default, yet for these recreations a precise arrangement is done. The residuals for the distinctive turbulence show factors were set to $10e^{-6}$ and the iteration max count to 2500. The reenactment procedure could likewise be ended or ceased if the CL or CD appeared to have balanced out appropriately.

Governing Equation

For this analysis, continuity equation, and momentum equation are solved using FLUENT. The equations are as follows:

$\frac{\partial \rho}{\partial t} + \nabla.(\rho u) = Sm$ Equation for conservation of mass

$\frac{\partial(\rho u)}{\partial t} + \nabla.(\rho u) = -\nabla p + \nabla.(\tau) + \rho g + F$ Conservation of momentum in an inertial frame

RESULTS

In this study, numerical investigations were performed. The tests were led at subsonic turbulent flow (Mach = 0.44) wind velocity (V) around NACA0012 airfoil at various various attack angle in the vicinity of 2° and 10° were estimation is simulated using turbulence models and the obtained results are compared with the literature survey of experimental data. The variations of lift coefficient CL and drag coefficient CD versus the AOA are given in figures, respectively. The values of Cl (Coefficient of lift) and Cd (Coefficient of drag) for different AOA, as obtained through FLUENT are presented below in tabular format.

Table 2: The Values of Cl and Cd for Different AOA

SN	AOA (^{0})	Cl	Cd
1	2	2.30E-01	8.94E-03
2	4	4.72E-01	9.09E-03
3	6	6.99E-01	1.25E-02
4	8	9.17E-01	1.68E-04
5	10	1.00E+00	3.12E-02

After post processing in FLUENT, following residual, Cd, Cl, and Cp plots, pressure and velocity contours were generated according to calculated data for different angle of attack ranging from 2 degrees to 10 degrees with an interval of 2 degree.

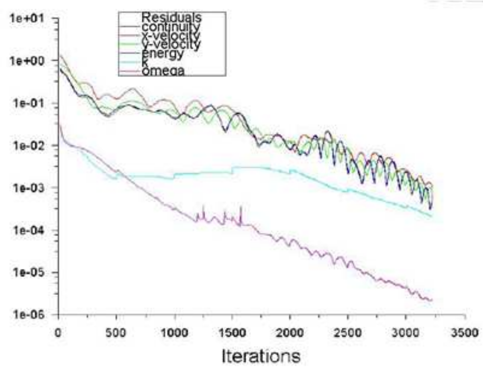

Figure 3.1: Scaled Residuals at 2 Degree of AOA

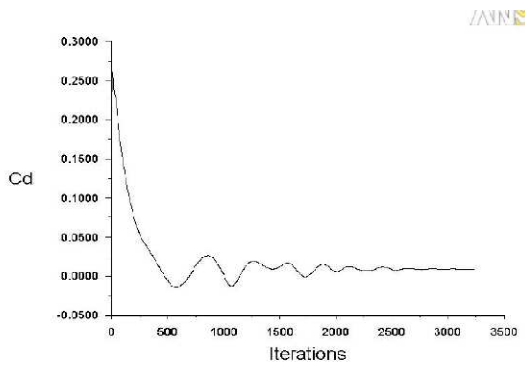

Figure 3.2: Drag Convergence History at 2 Degree of AOA

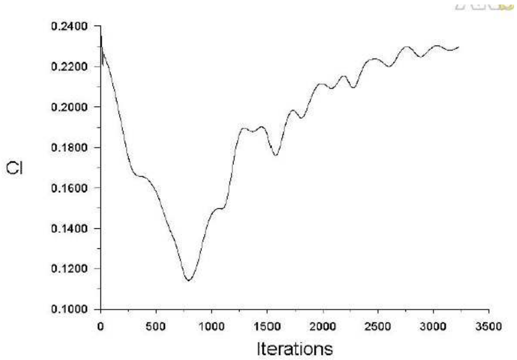

Figure 3.3: Lift Convergence History at 2 Degree of AOA

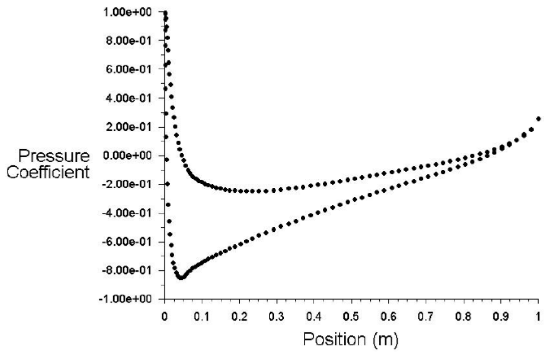

Figure 3.4: Pressure Coefficient at 2 Degree of AOA

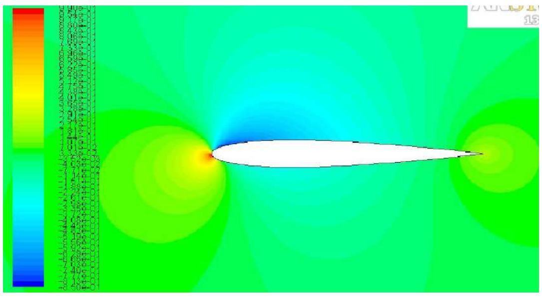

Figure 3.5: Contours of Pressure Coefficient at 2 Degree of AOA

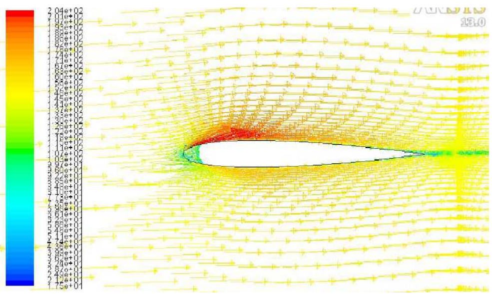

Figure 3.6: Velocity Vector Colored by Velocity Magnitude (m/s) at 2 Degree of AOA

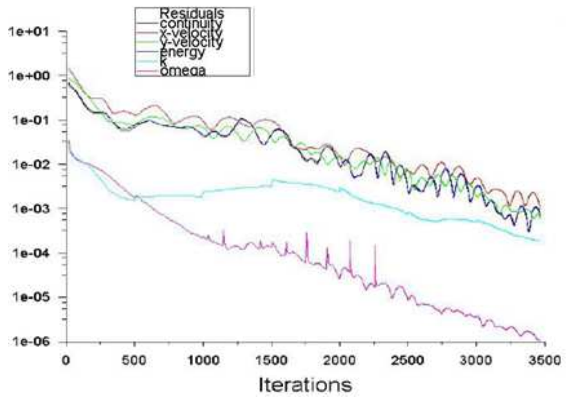

Figure 3.7: Scaled Residuals at 4 Degree of AOA

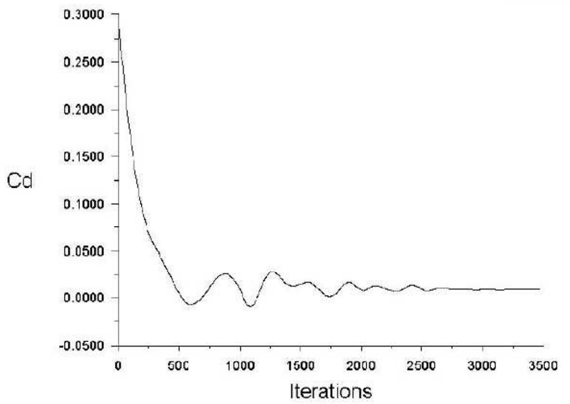

Figure 3.8: Drag Convergence History at 4 Degree of AOA

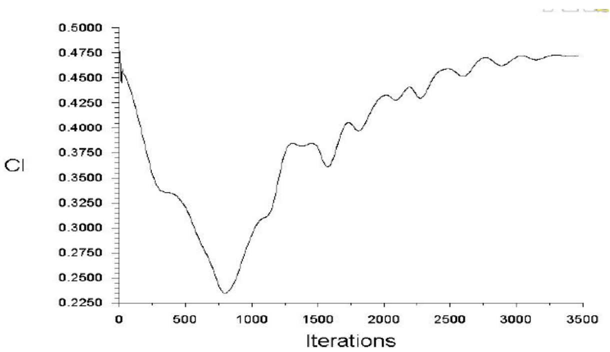

Figure 3.9: Lift Convergence History at 4 Degree of AOA

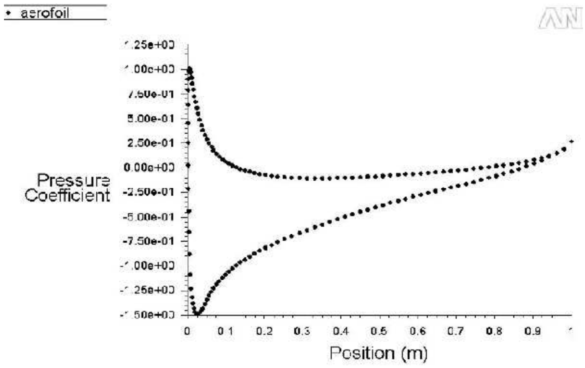

Figure 3.10: Pressure Coefficient at 4 Degree of AOA

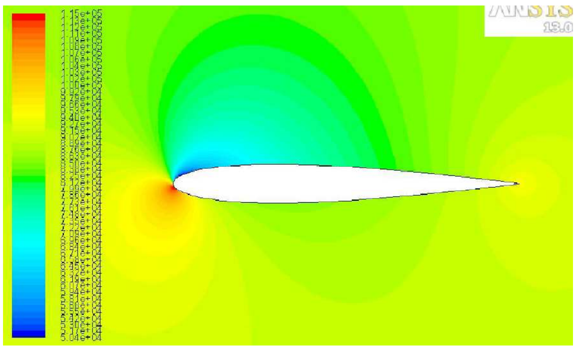

Figure 3.11: Contours of Pressure Coefficient at 4 Degree of AOA

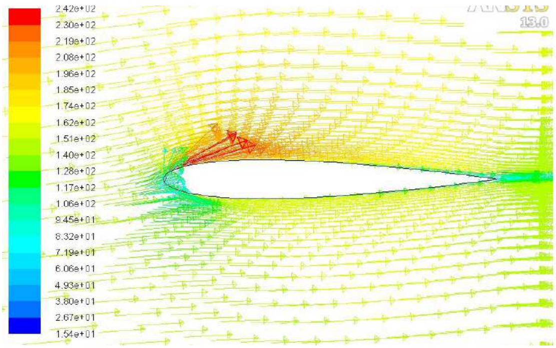

Figure 3.12: Velocity Vector Colored by Velocity Magnitude (m/s) at 4 Degree of AOA

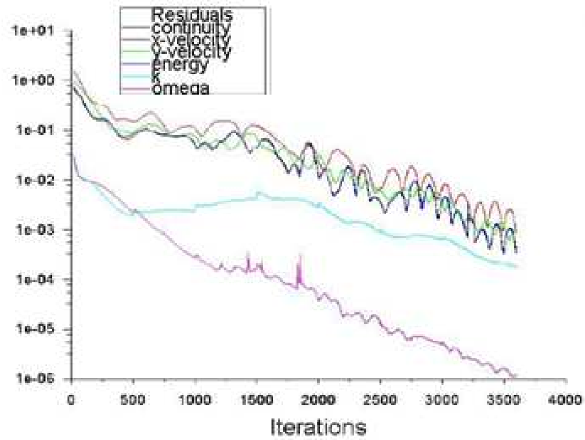

Figure 3.13: Scaled Residuals at 6 Degree of AOA

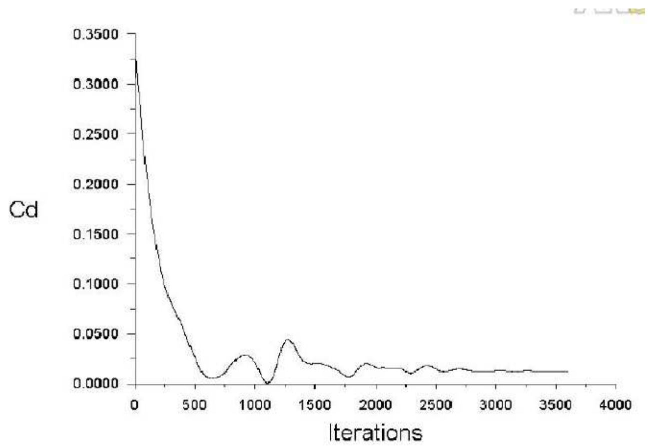

Figure 3.14: Drag Convergence History at 6 Degree of AOA

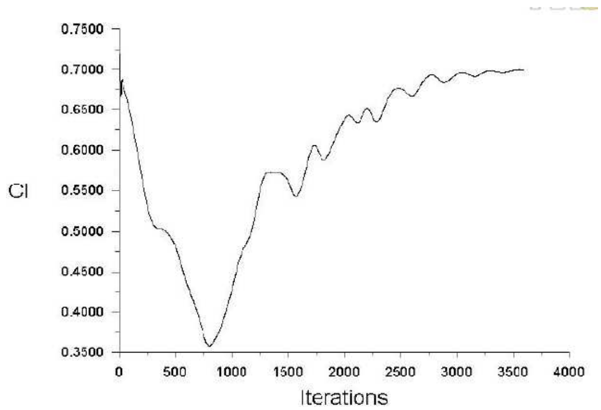

Figure 3.15: Lift Convergence History at 6 Degree of AOA

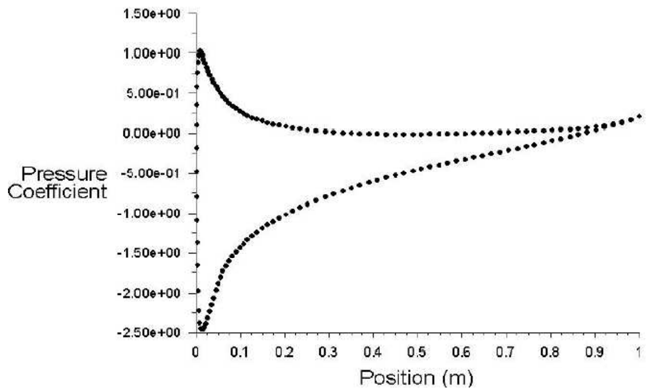

Figure 3.16: Pressure Coefficient at 6 Degree of AOA

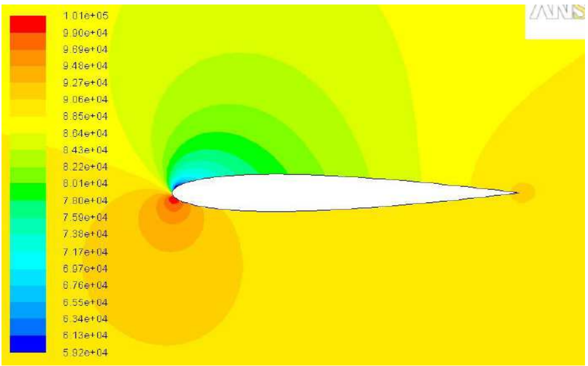

Figure 3.17: Contours of Pressure Coefficient at 6 Degree of AOA

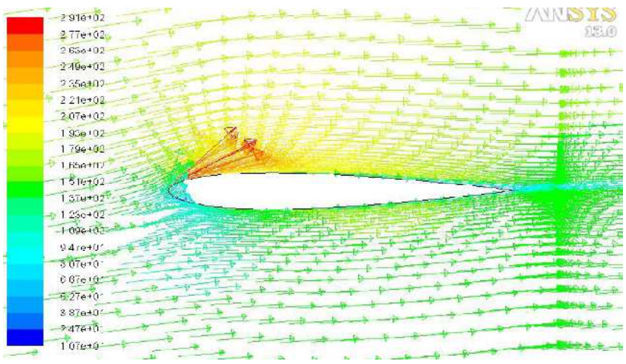

Figure 3.18: Velocity Vector Colored by Velocity Magnitude (m/s) at 6 Degree of AOA

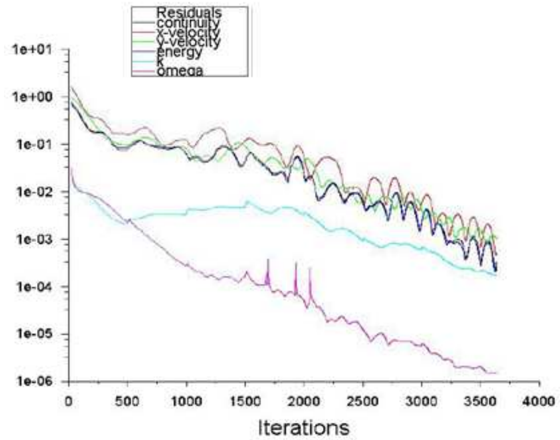

Figure 3.19: Scaled Residuals at 8 Degree of AOA

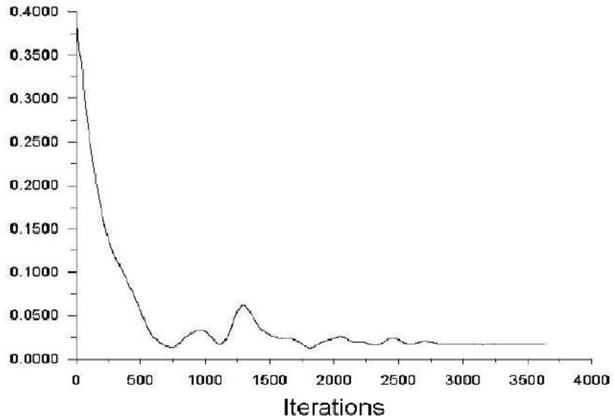

Figure 3.20: Drag Convergence History at 8 Degree of AOA

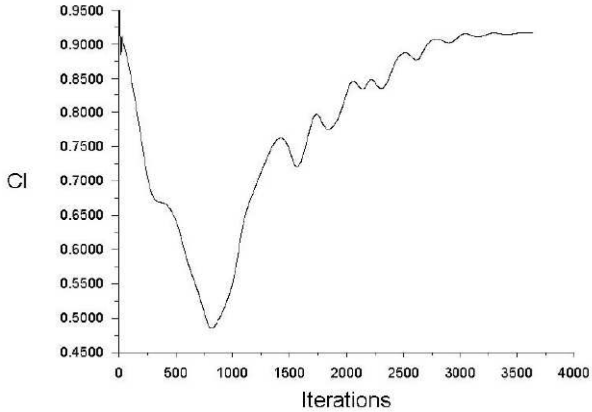

Figure 3.21: Lift Convergence History at 8 Degree of AOA

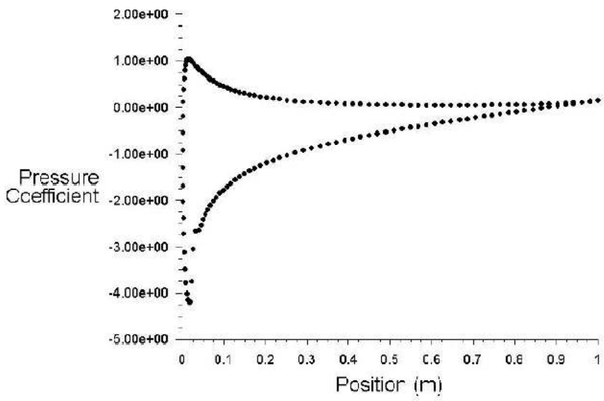

Figure 3.22. Pressure Coefficient at 8 Degree of AOA

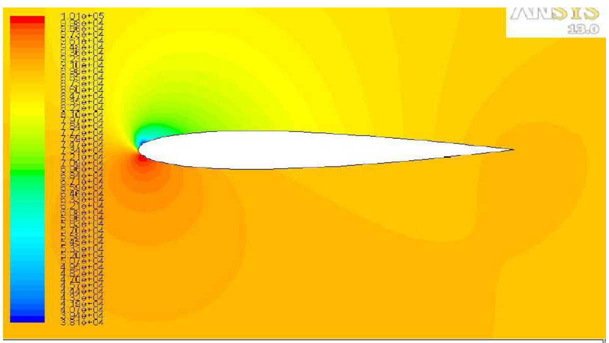

Figure 3.23: Contours of Pressure Coefficient at 8 Degree of AOA

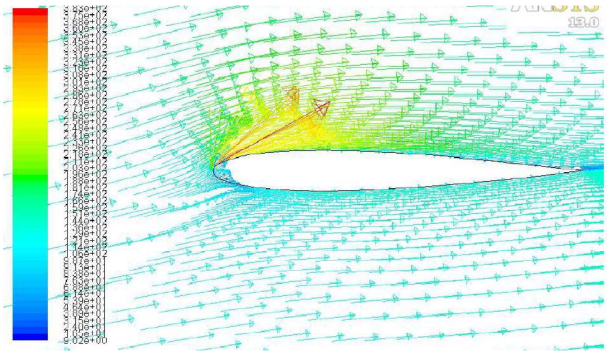

Figure 3.24: Velocity Vector Colored by Velocity Magnitude (m/s) at 8 Degree of AOA

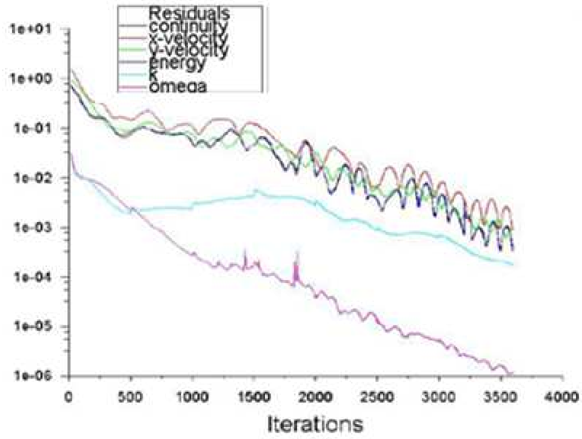

Figure 3.25: Scaled Residuals at 10 Degree of AOA

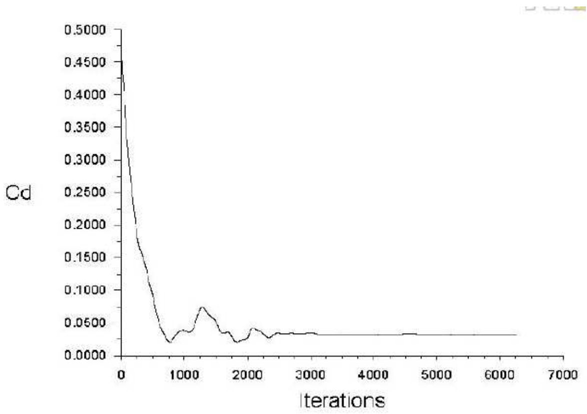

Figure 3.26: Drag Convergence History at 10 Degree of AOA

Figure 3.27: Lift Convergence History at 10 Degree of AOA

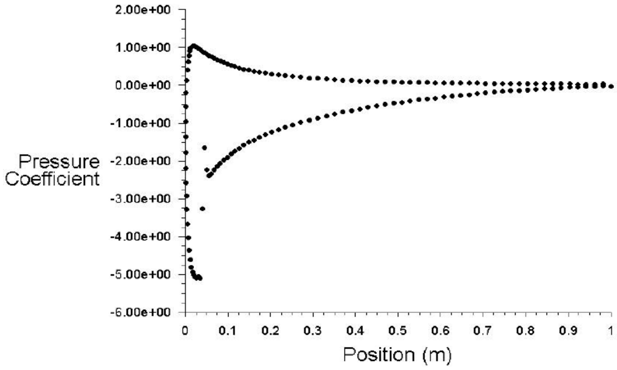

Figure 3.28: Pressure Coefficient at 10 Degree of AOA

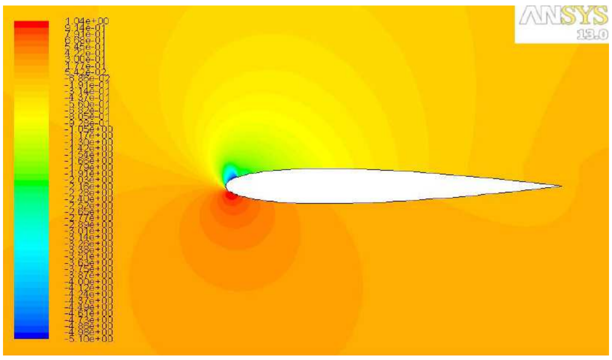

Figure 3.29: Contours of Pressure at 10 Degree of AOA

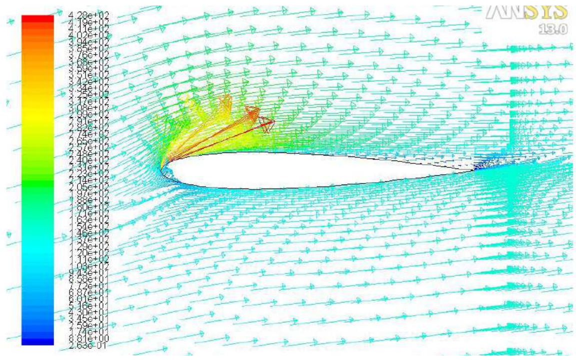

Figure 3.30: Velocity Vector Colored by Velocity Magnitude (m/s) at 10 Degree of AOA

CONCLUSIONS

In this research work, the precision of turbulence model of k-ω SST in the numerical analysis of flow over NACA0012 airfoil is explored utilizing CFD. The consequences of k-ω SST turbulence show is intended for aviation applications and offers fine outcomes for boundary layers being presented to the opposite weight. The drag coefficient anticipated by the turbulence models was more prominent than that got from the results. The computation comes about were given as takes after; coefficient Drag and coefficient lift expanded with expanding approach.

In this investigation lift and drag exhibitions of NACA0012 airfoil were performed. A FLUENT program was utilized to numerical estimations. Numerical and trial comes about were analyzed. The estimation comes about were given as takes after: Drag and lift coefficients expanded with expanding approach. Slow down was begun with 8° angle of attack. Lift coefficient diminished while; drag coefficient expanded. The ideal lift coefficient esteem was measured and figured at 10° The ideal airfoil execution was measured and ascertained at around 8° numerical investigation were demonstrated a decent outcomes and closeness.

REFERENCES

1. *Douvi C. Eleni, Tsavalos I. Athanasios and Margaris P. Dionissios, Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil.*
2. *Eleni c. Douvi, Athanasios I. Tsavalos, Dionissios P. Margaris, CFD calculations of the flow over a NACA0012 airfoil, Fluid Mechanics Laboratory (FML) University of Patras, Dept. of Mechanical Engineering and Aeronautics 26500 Patras, Greece.*
3. *Karna S. Patel, Saumil B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja, CFD Analysis of an Aerofoil, UVPCE, Ganpat University*
4. *Connell, J. R., "The Spectrum of Wind Speed Fluctuations Encountered by a Rotating Bladeof an Energy Conversion System," Solar Energy, Vol 29, No5, pp. 363–375, 1982.*
5. *Bertagnolio. F, Sørensen. N and Johansen. J, "Profile Catalogue for Airfoil Sections Based on 3D Computations" Tech. Rep. Risø-R-1581(EN), RISØ National Laboratory, Roskilde, 2006.*
6. *W. A. Timmer An overview of NACA 6-digit airfoil series characteristics with reference to airfoils for large wind turbine blades. Delft University Wind Energy Research Institute.*
7. *Naveen Janjanam, K. Raja Babu, S. S. Chowhan, Abid Khan, M. Teja, Comparison of NACA 23024 Aerofoil with and Without Vortex Generators Using CFD, International Journal of Mechanical Engineering and Technology, 8(5), 2017, pp. 556-566.*
8. *S. Jebarose Juliyana, Dr. J. Udaya Prakash, K. Karthik, P. Pallavi and M . Saleem, Design and analysis of NACA4420 wind Turbine aerofoil using CFD, International Journal of Mechanical Engineering and Technology (IJMET) Volume 8, Issue 6, June 2017, pp. 403–410.*
9. *Ranganayakulu Chennu, Pallavi Paturu, Development of heat transfer coefficient and friction factor correlations for offset fins using CFD, International Journal of Numerical methods for Heat & Fluid flow, Volume 21, Issue 8, 2011, pp. 935-951.*